Invention Journal of Research Technology in Engineering & Management (IJRTEM)
ISSN: 2455-3689
www.ijrtem.com Volume 2 Issue 8 ǁ August 2018 ǁ PP 53-59

Optimization of Design Parameters for Crane Hook Using Finite Element Analysis

Dipak A. Bhise[1], Prof. N.V. Deshpande[2]
[1](Department Mechanical Engineering, MPGI Nanded / SRTMU, Nanded, India
[2](Department Mechanical Engineering, MPGI Nanded / SRTNU, Nanded, India.

ABSTRACT : *The Crane hooks are very at risk segments that are regularly utilized for mechanical purposes. In this way such segments in an industry must be produced and composed in an approach to convey most extreme execution without failure. Failure of a crane hook essentially relies upon three central point i.e. measurement, material, overload. The undertaking is worried towards expanding the safe load by fluctuating the cross-sectional measurements of the four distinct segments and diverse materials. The chose areas are square, circle, and trapezoidal. The territory stays consistent while changing the measurements of the four unique segments. The crane hook is demonstrated utilizing catia programming. The pressure and life investigation is finished utilizing ANSYS 18.1 workbench. The ordinary worry along add up to misshaping, stress and life's according to the materials considered. It is discovered that trapezoidal cross segment yields most extreme load of 4000 KG to 5000 KG for steady cross segment zone among four cross segment.*

KEYWORDS: *Crane hook; Optimization; Finite element method; Total deformation.*

I. INTRODUCTION

A crane is a type of machine, generally equipped with a hoist rope, wire ropes or chains, and sheaves, that can be used both to lift and lower materials and to move them horizontally. It is mainly used for lifting heavy things and transporting them to other places. The device uses one or more simple machines to create mechanical advantage and thus move loads beyond the normal capability of a human. Cranes are commonly employed in the transport industry for the loading and unloading of freight, in the construction industry for the movement of materials, and in the manufacturing industry for the assembling of heavy equipment. The first known construction cranes were invented by the Ancient Greeks and were powered by men or beasts of burden, such as donkeys. These cranes were used for the construction of tall buildings. Larger cranes were later developed, employing the use of human tread wheels, permitting the lifting of heavier weights. In the High Middle Ages, harbor cranes were introduced to load and unload ships and assist with their construction some were built into stone towers for extra strength and stability. The earliest cranes were constructed from wood, but cast iron, iron and steel took over with the coming of the Industrial Revolution. The crane hook is demonstrated utilizing catia programming.

The pressure and life analysis is finished utilizing ANSYS 18.1 workbench. The typical worry along add up to twisting, stress and life's according to the materials considered is discovered that trapezoidal cross segment yields greatest load of 4000 KG to 5000 KG for steady cross segment region among four cross area. In this paper we will be able to define total deformation, stress etc.

II. LITERATURE SURVEY

Crane hook are highly significant component used for lifting the load with the help of chain or links. In the present paper a crane hook is purchased from the local market for Finite element analysis. The hook was tested on the UTM machine in tension to locate the area having maximum stress and to locate the yield point. The model of hook is prepared in CAE software having dimension and material similar to the crane hook which was purchased from the market. The results obtained were compared with theoretical analysis. Then cross section in which minimum stress induced for given load was modified through FEM. A. Gopichand [1] conveyed an advancement of plan parameters for crane hook utilizing Taguchi strategy. Crane hooks are one of the essential parts which are utilized to exchange material shaving substantial loads, for the most part in enterprises. Crane hooks are obligated segments subjected to failure because of focusing on amassing of overwhelming loads. The plan parameters for crane hook are territory of cross area, material and sweep of crane hook. In the present work advancement of outline parameters is completed utilizing Taguchi technique, add up to three parameters are considered with blended levels and L16 orthogonal exhibit is produced. The ideal mix of information parameters for least Vanishes stresses are resolved.

R. Uddanwadikerin [8] Crane hooks are highly liable components that are typically used for industrial purposes. Failure of a crane hook mainly depends on three major factors i.e. dimension, material. Its load carrying capacity is studied by varying the cross sections.

III. METHODOLOGY

The optimization of change in design parameters of crane hook with different material and its analysis is done through the CATIA and ANSYS software. Research methodology to employ in CATIA and ANSYS. CATIA: - CATIA means Computer-Aided Three-Dimensional Interactive Application, pronounced is a multi-platform software suite for computer-aided design, computer-aided manufacturing, computer-aided engineering, PLM and 3D, developed by the French company Dassault Systems. Its objective is to draw sketches and convert it into 3D space and apply constraints and relations [3]. ANSYS: - ANSYS a result of SASI (System Analysis Simulation and Integrating). It is a world's driving programming which keep running on CAE and FEM. It is broadly utilized by fashioner's investigation in businesses, for example, aviation, car, producing, atomic, gadgets, biomedical, and substantially more. In ANSYS, the rudiments of FEA ideas, demonstrating and the investigating of designing issue utilizing ANSYS workbench. Furthermore, portray of significance apparatuses and ideas given at whatever point required. this following reproduction surges of ANSYS. Ansys software is divide into characterizes analysis. Structural Analysis is divided into two types Static Structural Analysis and Explicit Dynamics Analysis [4].

IV. FINITE ELEMENT ANALYSIS

The Finite Element Method (FEM) or Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solutions of these problems generally require the solution to boundary value problems for partial differential equations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain. To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variation methods from the calculus of variations to approximate a solution by minimizing an associated error function [5].

General Procedure to Conduct Finite Elements Analysis:

- Set the type of analysis to use.
- Create model
- Define the elements type
- Divide the elements types
- Divide the given geometry into nodes and elements
- Apply material properties and boundary conditions
- Drive elements matrices and equations
- Solve the unknown parameters at nodes
- Interpret the results [8].

Static Structural Analysis and Explicit Dynamic Analysis for Steel: A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. A static structural load can be performed using the ANSYS, Samcef, or ABAQUS solver. Explicit Dynamics is a transient explicit dynamics Workbench application that can perform a variety of engineering simulations, including the modeling of nonlinear dynamic behavior of solids, fluids, gases and their interaction [6]. In this paper we have considering four modal and five materials afterwards comparing each and each modal for each and every material. Ansys 18.1 workbench is used to find stress and life. According to static structural analysis and explicit dynamic analysis all results are tabulated and compared. We proposed that to expanding the safe load from 4000kg to 5000kg, modal namely trapezoidal and square cross section is better for steel material [7].

The explicit dynamic analysis is one of the critical investigations in ANSYS work bench.

1) For Modal: 01
   a) Trapezoidal and Circle Cross Section
      i. From Static Structural Analysis:
         Total deformation =1.4193mm.

         Equivalent stress=289.39mpa.

         Maximum principle stress=299.36mpa.

         Life= 7353psi.

      ii. From Explicit Dynamic Analysis:
         Total deformation =1.2177mm.

         Equivalent stress=1277.4mpa.

2) For Modal: 02
   b) Trapezoidal and Square Cross Section

      i. From Static Structural Analysis:
         Total deformation =1.4426mm.

         Equivalent stress=290.02mpa.

         Maximum principle stress=290.02mpa.

         Life= 7305psi.

      ii. From Explicit Dynamic Analysis:
         Total deformation =1.2178mm.

         Equivalent stress=1478.3mpa.

3) For Modal: 03
   c) I Section and Circle Cross Section

      i. From Static Structural Analysis:
         Total deformation =1.4975mm.

         Equivalent stress=303.26mpa.

         Maximum principle stress =305.86mpa.

         Life= 6363psi.

      ii. From Explicit Dynamic Analysis:
         Total deformation =1.2176mm.

         Equivalent stress=1776.3mpa.

4) For Modal: 04
   d) I Section and Square Cross Section

      i. From Static Structural Analysis:
         Total deformation =1.5284mm.

         Equivalent stress=304.82mpa.

         Maximum principle stress=307.53mpa.

Life= 6263psi.

ii. From Explicit Dynamic Analysis:
Total deformation =1.2177mm.

Equivalent stress=2010.6mpa.

V. RESULT AND GRAPH

For Modal 2 Trapezoidal and Square Cross Section Static Structural Analysis.

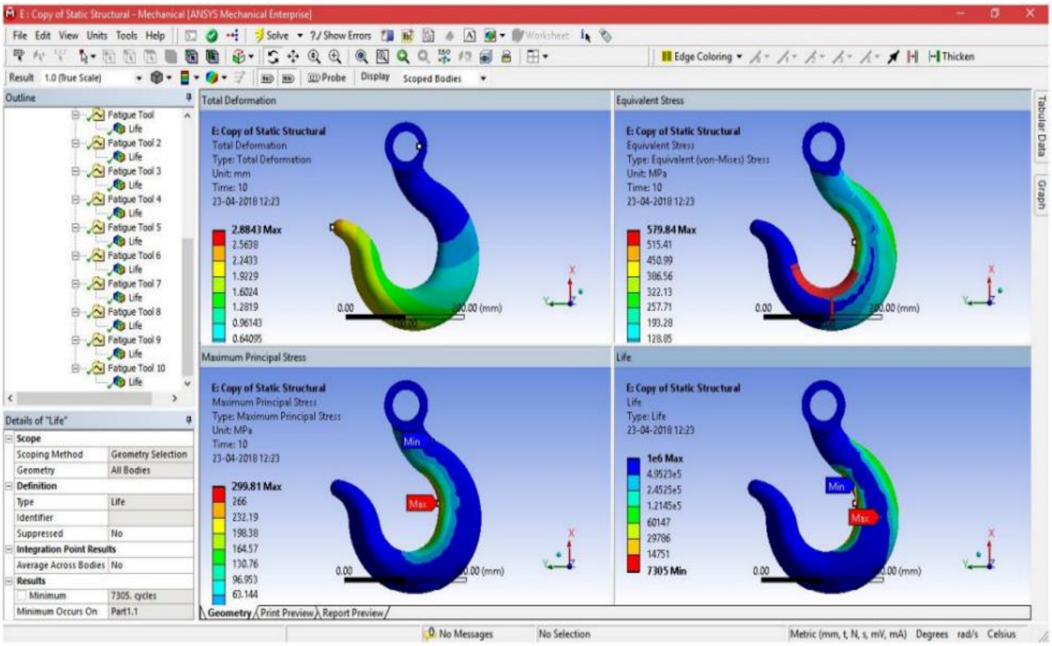

Figure 1. Total deformation, Equivalent stress, Life and max. principle stress of rod.

Table 1: Total Deformation and Equivalent Stress

Load	For Static Structural Analysis					For Static Structural Analysis				
	Total Deformation					Equivalent Stress				
	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel
4100	1.183	3.3276	2.4603	2.1506	1.2225	237.81	238.26	238.93	237.14	237.62
4200	1.2118	3.4087	2.5202	2.2031	1.2523	243.61	244.07	244.76	242.92	243.42
4300	1.2407	3.4899	2.5803	2.2555	1.2821	249.42	249.89	250.59	248.71	249.21
4400	1.2695	3.571	2.6403	2.308	1.312	255.22	255.7	256.42	254.49	255.01
4500	1.2984	3.6522	2.7003	2.3604	1.3418	261.02	261.51	262.25	260.28	260.81
4600	1.3272	3.7334	2.7603	2.4129	1.3716	266.82	267.32	268.07	266.06	266.6
4700	1.3561	3.8145	2.8203	2.4653	1.4014	272.62	273.13	273.9	271.84	272.4
4800	1.3849	3.8957	2.8803	2.5178	1.4312	278.42	278.94	279.73	277.63	278.19
4900	1.4138	3.9768	2.9403	2.5702	1.4611	284.22	284.75	285.56	283.41	283.99
5000	1.4426	4.058	3.0003	2.6227	1.4909	290.02	290.57	291.38	289.19	289.78

Table 2: Max. Principle Stress and Life

	For Static Structural Analysis					For Static Structural Analysis				
	Max Principle Stress					Life				
Load	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel
4100	237.81	246.56	247.64	244.79	245.54	13924	5918	5702	6300	6133
4200	243.61	252.57	253.68	250.76	251.53	12830	4584	4486	4752	4679
4300	249.42	258.58	259.72	256.73	257.52	11837	3822	3739	3966	3903
4400	255.22	264.60	265.76	262.70	263.51	10940	3187	3116	3310	3256
4500	261.02	270.61	271.8	268.67	269.50	10130	2657	2597	2762	2716
4600	266.82	276.62	277.84	274.64	275.49	9453	2216	2164	2305	2266
4700	272.62	282.64	283.88	280.61	281.48	8844	1847	1804	1923	1890
4800	278.42	288.65	289.92	286.58	287.47	8287	0	0	0	0
4900	284.22	294.66	295.96	292.55	293.45	7775	0	0	0	0
5000	290.02	300.68	302	298.53	299.44	7305	0	0	0	0

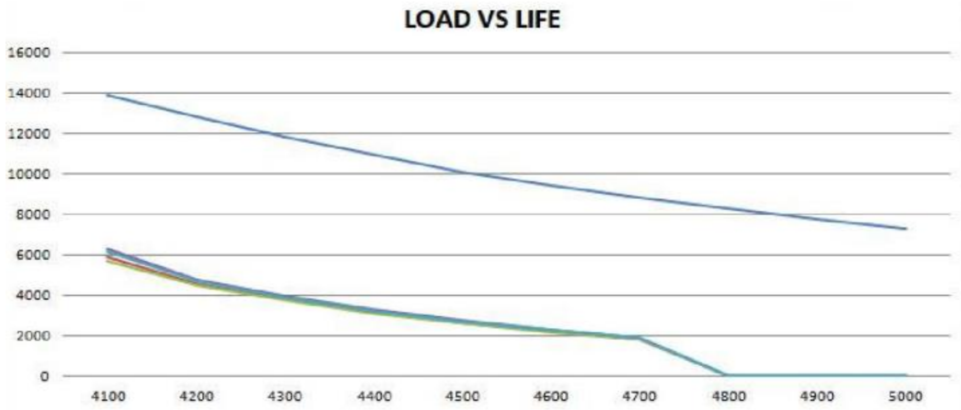

Figure 2. Static Structural Analysis Graph load verses Life.

For Modal 2 Trapezoidal and Square Cross Section Explicit Dynamic Analysis

Results for Steel; if ROD made up of Steel

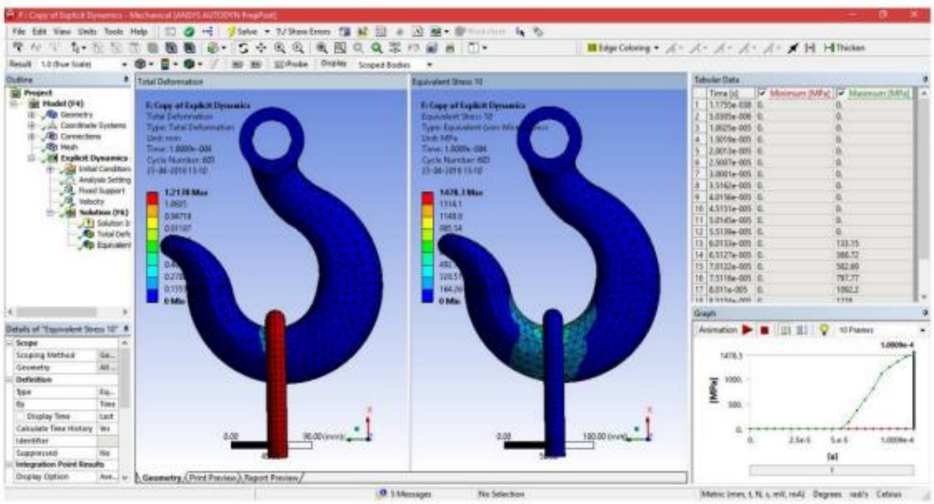

Figure 3. Total Deformation, Equivalent Stress, of Rod

Load	For Explicit Dynamics Analysis					For Explicit Dynamics Analysis				
	Total Deformation					Equivalent Stress				
	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel	Steel	Al Alloy	Ti Alloy	Ci	Low Alloy Steel
	0.0000	0.0000	0.0000	0.0000	0.0000	0	0	0	0	0
	0.0143	0.0143	0.0146	0.0140	0.0148	0	0	0	0	0
4100	0.0575	0.0579	0.0589	0.0591	0.0578	0	0	0	0	0
4150	0.11599	0.11651	0.11613	0.11713	0.11735	0	0	0	0	0
4200	0.17517	0.17587	0.17603	0.17587	0.17565	0	0	0	0	0
4250	0.23506	0.23593	0.23665	0.23531	0.23669	0	0	0	0	0
4300	0.29565	0.29671	0.29594	0.29544	0.29639	0	0	0	0	0
4350	0.35901	0.3582	0.35799	0.35905	0.35887	0	0	0	0	0
4400	0.42104	0.4204	0.42076	0.42062	0.41998	0	0	0	0	0
4450	0.48378	0.48332	0.48213	0.48287	0.48391	0	0	0	0	0
4500	0.54723	0.54694	0.54633	0.54583	0.5464	0	0	0	0	0
4550	0.61138	0.61128	0.61125	0.60948	0.60958	0	0	0	0	0
4600	0.67624	0.67632	0.6747	0.67676	0.67567	133.15	64.368	70.183	58.873	123.79
4650	0.74181	0.74208	0.74105	0.74184	0.74025	368.72	152.02	207	191.59	351.79
4700	0.80809	0.80856	0.80812	0.80762	0.80778	582.69	251.89	325.26	312.65	563.9
4750	0.87508	0.87356	0.87365	0.87409	0.87376	797.77	340.01	448.34	443.52	767.33
4800	0.94278	0.94144	0.94215	0.94126	0.94273	1092.2	470.17	613.77	610.5	1055.6
4850	1.011	1.0099	1.0112	1.012	1.0099	1228	537.97	702.67	716.68	1186.8
4900	1.0796	1.0786	1.0783	1.0801	1.0798	1342.2	576.35	756.56	793.4	1302.1
4950	1.1485	1.1477	1.148	1.1485	1.1477	1432.7	600.46	791.93	835	1389.5
5000	1.2178	1.2172	1.2182	1.2173	1.2183	1478.3	616.6	813.8	908.51	1436.1

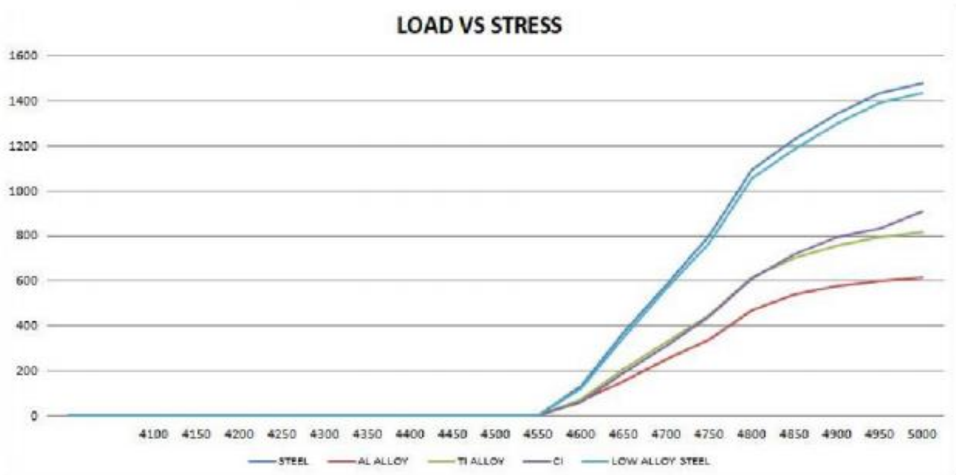

Figure 3. Total Deformation and Equivalent Stress

VI. CONCLUSION

The task is worried towards expanding the safe load by changing the cross sectional measurements of the four unique areas and diverse materials as for the segments are square, circle, and trapezoidal. The crane hook is demonstrated utilizing catia programming. The pressure and life analysis is finished utilizing ANSYS 18.1 workbench. The typical worry along add up to twisting, stress and life's according to the materials considered. It is discovered that trapezoidal cross segment yields greatest load of 4000 KG to 5000 KG for steady cross segment region among four cross area. From the static structural analysis, the life of the wrench hook is more for the cast press for same cross area, yet when it think about by cross segment of modular 1,2,3 and 4 the modular 1 (circle and trapezoidal) is high. Yet, when we are contrasting with deference with push modular 3 (circle and I segment trapezoidal). From the explicit dynamic analysis the worry of the wrench hook is more for the steel for same cross area, however when it think about by cross segment of modular 1,2,3 and 4 the modular 2 (square and trapezoidal) is high. In any case, when we are contrasting with deference with push modular 4 (square and I area trapezoidal) steel is high.

REFERENCES

[1]. Optimization of design parameters for crane hook using Taguchi method. **Gopichand, A and Lakshmi, RVS and Maheshkrishna, B.** 12, s.l. : Int. J. Innov. Res. Sci. Eng. Technol., Vol. 2. 2319-8753.

[2]. Study of Crane Hook Having Trapezoidal Section by Finite Element. **Santosh Shaun, Ritesh Dewangan, Manas Patnaik, Narendra Yadav.** 2779-2781, s.l. : International Journal of Modern Engineering Research , July-Aug 2012, Vol. 2.

[3]. **R.S.Khurmi.** Strength of materials. 23rd Edition. 2009.
[4]. Finite Element Analysis of the Tower Crane. **Ismail Gerdemeli, Serpil Kurt, Okan Deliktas.** s.l. : 14th International Research/Expert Conference, September 2010.
[5]. Finite element approach to 3D Modeling and stress analysis of crane lifting hooks. **Derya Ozer, A.Burak Erdil, Ismail Gerdemeli.** s.l. : International Research/Expert Conference TMT , Hammamet, Tunisia, September, 2007.
6. "Finite Element Analysis using ANSYS 11.0. **Paleti Srinivas, K.C.Sambana, Rajeshkumar Datti.** s.l. : PHI Learning Pvt. Ltd. New Delhi.
[7] . Modeling of a crane hook wearand stress analysis. Transport Means 2010. **Narvydas, E.** 161-164 , s.l. : Proceedings of the 14th international, Oct 21-22,2010.
[8]. Stress Analysis of Crane Hook and Validation by Photo-Elasticity. **Uddanwadiker, R.** s.l. : Engineering, 2011, Vol. 3.

Dipak A. Bhise, Optimization of Design Parameters for Crane Hook Using Finite Element Analysis. Invention Journal of Research Technology in Engineering & Management (IJRTEM), 2(8), 53-59. Retrieved August 24, 2018, from www.ijrtem.com.